Advances in Engineering Software 000 (2017) 1–9

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Numerical assessment of fatigue design curve of welded T-joint improved by high-frequency mechanical impact (HFMI) treatment

Caiyan Deng[a,b], Yaru Niu[a,b], Baoming Gong[a,b,*], Yong Liu[a,b], Dongpo Wang[a,b]

[a] Department of Materials Science and Engineering, Tianjin University, Tianjin 300354, China
[b] Tianjin Key Laboratory of Advanced Joining Technology, Tianjin 300354, China

ARTICLE INFO

Article history:
Received 17 March 2017
Revised 3 May 2017
Accepted 28 June 2017
Available online xxx

Keywords:
Structural hot spot stress approach
High frequency mechanical impact
Fatigue design
Thickness effect
Material strength

ABSTRACT

In the paper, the fatigue performances of as-welded T-joint and T-joint improved by high frequency mechanical impact (HFMI) were numerically investigated using structural hot spot stress approaches: linear surface extrapolation (LSE) and through thickness at the weld toe (TTWT). The effects of main plate thickness and material strength for HFMI-treated joints were investigated. The results showed that the TTWT method was more effective to study the effect of thickness on T-joints improved by HFMI treatment than LSE method. For as-welded T-joints, the thickness correction exponent $n = 0.04$ was obtained when the attachment plate thickness was set as constant. For HFMI-treated T-joints, a reverse thickness effect was observed with negative thickness correction exponents, and the thickness correction exponents increased with material strength. In addition, the adoption of S–N slope varying with yield strength was proven to be more proper for HFMI improvement assessment.

1. Introduction

Welded structures subjected to cyclic loading are sensitive to fatigue damage, and fatigue crack tends to initiate from weld toe because of the local stress concentration and welding residual tensile stress. Recently, high-frequency mechanical impact (HFMI) treatment has been widely used as a post-weld treatment, which utilizes powerful energy to drive the needle to impact the object material at a high frequency. Therefore, the effectiveness of HFMI treatment is attributed to the smoothing of local stress concentration, refinement of microstructure and generation of local compressive residual stress at the weld toe [1–12]. Some works [13–15] demonstrate that the improvement degree of HFMI treatment is better than commonly applied methods recommended by [16].

Previous studies [17–21] indicate that the fatigue performances of HFMI-treated welded joints are quite different from the as-welded ones. According to IIW [22], the slope value of S-N curves for welded joints of steel and aluminum is designated as $m = 3$, independent of material strength and stress range. However, the related works demonstrate that the slope value of S-N curve for post-weld treated joints is gentler than that of as-welded joints [17–19], and the degree of improvement varies with material strength [20]. The guidelines proposed by IIW [22] roughly adjusted the strength levels of specific joints. Recently, Yildirim and Marquis [18] provided an overview of 228 published data on fatigue performance of HFMI-treated joints and a new fatigue design recommendation was suggested that by choosing $f_y = 355$ MPa as a reference, approximately 12.5% increase in fatigue strength for every 200 MPa increase in yield strength with an assumed S-N slope $m = 5$ for HFMI-treated welded joints. Wang et al. [19] also studied the effect of HFMI treatment on fatigue performances of welded joints, and reported that the slope values of S-N curves of HFMI-treated joints may vary from 6.3 to 23 and $m = 10$ was thus proposed for fatigue design of UPT welded joints. Moreover, the thickness effect on HFMI-treated joints is not trivial. According to IIW [22], the effect of plate thickness on fatigue strength should be taken into consideration in cases fatigue crack initiates from weld toe. The thickness reduction factor $f(t)$ for as-welded joints is given as follows:

$$f(t) = \left(\frac{t_{ref}}{t_{eff}}\right)^n \tag{1}$$

where $t_{ref} = 25$ mm, n is the thickness correction exponent dependent on the effective thickness t_{eff} and the joint category.

Numerous experiments of HFMI treatment, fatigue tests and residual stress measurements are both time-consuming and expensive, it is hard to obtain full measurement of stress field. Therefore, dynamic explicit finite element method (FEM) has been widely used to simulate the process of HFMI treatment [23–27]. Yang *et*

---

[*] Corresponding author at:. Tianjin Key Laboratory of Advanced Joining Technology, Department of Materials Science and Engineering, Tianjin University, Tianjin 300354, China.

E-mail address: gongbm@tju.edu.cn (B. Gong).

http://dx.doi.org/10.1016/j.advengsoft.2017.06.017

al [23]. investigated the fatigue performance of AISI304 stainless steel treated by ultrasonic impact treatment. The finite element model was established using Johnson-Cook material model to get the residual stress. Kuilin and Yoichi [26] numerically studied the effect of ultrasonic impact treatment on welded joints. A good agreement was observed between the predicted and measured results. The numerical simulation method has been proved to be an effective method, and the validity of numerical simulation method has been demonstrated [27].

Although noticeable progress on fatigue assessment of welded joints improved by post-weld treatment methods has been achieved, systematic investigation for different material strengths and thickness effects is still under development. In this paper, the numerical simulation method was adopted to study the fatigue performance of HFMI-treated fillet T-joint using structural hot spot stress approaches, i.e., linear surface extrapolation (LSE) and through thickness at the weld toe (TTWT). Four kinds of main plate thicknesses: 8 mm, 16 mm, 25 mm and 35 mm, were chosen to take the effect of main plate thickness into consideration. In addition, materials with different steel grades (20 steel, AISI 1006, 45 steel and AISI 2205) were employed due to the sensitivity of fatigue performance to material strength for HFMI-treated joints. At first, the process of HFMI treatment was simulated; Secondly, external load was applied to the HFMI-treated welded joints and the final stress fields were used to evaluate the structural hot spot stress. The aim of this work is to evaluate the thickness effect for HFMI-treated joints with different yield strengths using structural hot spot stress approaches.

2. Numerical modeling

2.1. Finite element model of HFMI treatment

The three-dimensional FEA model of HFMI-treated T-joint, axial section of the T-joint and needles along the weld direction are shown in Fig. 1. Half of T-joint is taken into consideration by virtue of the symmetry. A portion of T-joint is employed with the width of 5 mm and the length of 50 mm. The main plate thicknesses t used in this paper are 8 mm, 16 mm, 25 mm and 35 mm, the attachment plate thickness is set as a constant 8 mm. The leg lengths of the joints are set as 6.4 mm and the radius of the weld toe is assumed to be 1.5 mm. All dimensions apart from the main plate thickness are essentially the same. Both the distances between the first needle and the target model and the interval of the adjacent needles along the impact direction and the weld direction are set as 0.2 mm. It is an optimal compromise between the computing time and the sufficient overlap of the impacts. A high-speed video camera was used by Deng et al. [28] to determine the impact velocity during HFMI process. Accordingly, the impact velocity is set as 3.34 m/s with a coulomb friction coefficient of 0.3. The paths to extract the residual stress are plotted in Fig. 1(b).

To discretize the target model, three-dimensional 8-node hexahedral solid elements with reduced integration and hourglass control (C3D8R) are employed. To improve the accuracy of simulation and reduce the consumption of time, local refined mesh is employed. The mesh near the contact surface of weld toe is refined with the minimum element size of $50\,\mu m \times 50\,\mu m \times 15\,\mu m$ and a biased mesh is adopted in the thickness direction. The impact needle is set as shell with a diameter of 3 mm and it is meshed using the linear quadrilateral elements (S4R). In the dynamic simulation of HFMI process, the needles are assumed as rigid bodies and the rotational degrees of freedom of the needles are restricted to ensure the stability of the impact process.

Table 1
Constitutive model parameters.

Material	A (MPa)	B (MPa)	C	p	q
20 steel	258	329	0.0323	1.05	0.235
AISI 1006	350	275	0.022	1.0	0.36
45 steel	507	320	0.064	1.06	0.28
AISI 2205	622	785.25	0.035	0.1515	0.5046

2.2. Johnson–Cook constitutive model

In this article, steels with different yield strengths (20 steel, AISI 1006, 45 steel, AISI 2205) are included to take into account the effect of material strength. According to the high speed and nonlinear characteristics of HFMI treatment, the strain rate must be taken into consideration. Johnson-Cook constitutive model which considers the strain rate and work hardening is selected.

$$\sigma = (A + B\varepsilon^{q})\left(1 + C\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\left[1 - \left(\frac{T - T_r}{T_m - T_r}\right)^{p}\right] \tag{2}$$

where σ means the von Mises stress, and A, B and C represent the initial yield strength, the strain hardening coefficient and strain rate sensitivity, respectively. The parameters p and q mean the thermal softening effect and the strain hardening role. The corresponding Johnson-Cook parameters are given in Table 1 [29–32]. The other material parameters used in simulation are set to be the same: Young's modulus 210 GPa, material density 7900 kg/m^3 and Poisson's ratio 0.3.

2.3. Structural hot spot stress approaches

Structural hot spot stress approach avoids the defects associated with the nominal stress approach, and is computationally less demanding than fracture mechanics methods. Structural hot spot stress takes into consideration the dimensions and stress concentrating effects of the detail at the anticipated crack initiation site while excluding the local non-linear stress peak caused by the notch at the weld toe. Recently, there is growing interest in the structural hot spot stress for welded joint assessment [33–35]. According to Niemi et al. [36,37], linear surface extrapolation (LSE) determines the structural hot spot stress based on the reference points located in front of the weld toe and extrapolation:

$$\sigma_{hs} = 1.67_{0.4t} - 0.67_{1.0t} \tag{3}$$

where t represents the thickness of main plate, $0.4t$ and $1.0t$ distances from the weld toe are used as shown in Fig. 2(a) [22].

In the through thickness at the weld toe (TTWT) procedure, the structural hot spot stress is calculated in terms of cross-section stress in the weld toe as shown in Fig. 2(b) [22]. The stress distribution is non-linear including membrane force (σ_{m}), bending force (σ_{b}) and nonlinear stress peak (σ_{lnp}), a linear stress field can be obtained by ignoring the nonlinear stress peak. The hot spot stress can be computed by Eqs. (4)–(6) [33,38].

$$\sigma_m = \frac{1}{t}\int_0^t \sigma_x(y)\cdot dy \tag{4}$$

$$\sigma_m \cdot \frac{t^2}{2} + \sigma_b \cdot \frac{t^2}{6} = \int_0^t \sigma_x(y)\cdot y \cdot dy \tag{5}$$

$$\sigma_{hs} = \sigma_m + \sigma_b \tag{6}$$

3. Residual stress distributions after external loading

In the Section, the residual stress distributions after external loading for HFMI-treated T-joint are determined numerically to assess the fatigue performance by structural hot spot stress approaches.

Please cite this article as: C. Deng et al., Numerical assessment of fatigue design curve of welded T-joint improved by high-frequency mechanical impact (HFMI) treatment, Advances in Engineering Software (2017), http://dx.doi.org/10.1016/j.advengsoft.2017.06.017